Application of OpenFOAM in Product Development Phase

Takeharu Kawamura*  Xiroyuki Shiozawa*  Masahiro Sano*  Takashi Yoda*
Toyohito Seino*  Hiromi kitsunoezu*  Misae Hagiwara**  Hiroshi Fujikawa**
Takumi Sakamoto**  Eikyuu Kin**  Yingpei Yi**

Abstract

The shape and the specifications of automotive products continue to be diversified in response to the expansion of market demands. This required significant reduction in program development time. As a result the demand for Computer-Aided Engineering (CAE) continues to increase. As CAE models become bigger in size, we can consider to utilize higher performance computer, or to use outside resource such as cloud-based computing technology, which requires in most cases to add more software solver licenses. In either case, license cost of the commercial software is very expensive. Calsonic Kansei has considered utilization of open source CFD software named OpenFOAM. In this report correlation study between OpenFOAM and experiment result is presented during the product development for Duct, HVAC, Defroster, Blower and Motor Fan applications. The study shows good agreement between the physical and the simulation results.

Key Words: CAE / CFD / OpenFOAM / Simulation

1. Introduction

Lately, CAE has been recognized as a necessary tool for efficient product development and has become a commonplace in manufacturing industry fields. Since CPU cost decreases with improved hardware performances, computers capable of processing CAE software may easily be introduced in development operations. In current computers, processing speed has been still increasing, proving the Moore’s law through 50 years (as of 2015). In our company, the CAE needs are expanding toward faster and larger-scale simulation on more complex analysis models. To fulfill these needs, however, large expenses are continuously incurred for upgrading software licenses. Furthermore, the license fees tend to increase in these years. On this account, solutions are required to realize large scale simulation with small expenses.

This report introduces a solution with application of open source CFD software named OpenFOAM.

2. Outline of OpenFOAM and Application Requirement

OpenFOAM is globally renowned open source CFD software which was developed at Gosman Laboratory in Imperial College (United Kingdom), known as a developer of STAR-CD. Despite the renown, there are several correlation results with simple shapes but only few with complex shapes as in actual products. Thus, this software has hardly been used for product development in Japanese manufacturing industry because thorough correlation is still required. In recent years, automotive makers in Japan have shown study reports of OpenFOAM application but most of them have not put it into practical use in product development (1)(2). Taking this matter into account, we studied correlation among OpenFOAM calculation results, bench test results, and calculation results of the currently used commercial software (software S and F) on the subject of products under development with CAE analyses. As a result, we verified that OpenFOAM can take the place of the currently used commercial software.

3. Correlation in Bent Pipe Internal Flow

Internal flow is mostly simulated for fluid analyses in our company. Thus, we firstly studied correlation of fluid flow in a simple conduit shape. The following
explains study results with a square pipe that has a bent portion (3).

### 3.1. Analysis Model and Condition

With OpenFOAM, airflow was simulated in the bent pipe shown in Fig. 1, and the result was compared with those derived from bench tests and the commercial software. In this analysis, the conditions were specified as follows: simulating incompressible stationary flow, specifying Reynolds number of approximately 40,000 in turbulent flow field, and using SST $k-\omega$ turbulent model.

### 3.2. Analysis Result

As basic fluid behavior, airflow velocity distribution is parabolic at the SECT 1 in Fig. 1, whereas the maximum velocity shifts inward at the SECT 2 because secondary vortex is caused by outward airflow due to the pipe curvature. At the SECT 3, the secondary vortex becomes even more noticeable. Fig. 2 shows the analyses and test results of airflow velocity distribution at the cross-section centers and velocity contours in each cross-section. In addition, the Y-axis corresponds to each arrow shown in Fig. 1. These results demonstrate that airflow distribution characteristics in OpenFOAM correlate with the test result. The absolute velocity at the SECT 3 partially deviates from the test results. However, the deviation also occurs at the same part in the commercial software. Hence, this deduces that the airflow velocity distribution in OpenFOAM qualitatively correlates with the test results.

<table>
<thead>
<tr>
<th>SECT 1</th>
<th>SECT 2</th>
<th>SECT 3</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="#" alt="Velocity distribution" /></td>
<td><img src="#" alt="Velocity contour" /></td>
<td><img src="#" alt="Velocity contour" /></td>
</tr>
</tbody>
</table>

Fig. 2 Velocity Profile

### 4. Application to Duct Airflow Resistance Analysis

Secondly, the application study was conducted on a duct product applied in climate systems. This product is mainly installed inside the instrument panel to guide airflow from the blower to the cabin. Since its installation space is severely restricted, CAE analyses are utilized in designing processes to verify the air pressure loss and the airflow distribution rate in each duct. In this study, a vent duct was used to verify that OpenFOAM is applicable in the airflow resistance analysis.

#### 4.1. Test Method

In the duct unit test, static pressure (total pressure) was measured in chambers placed at the duct inlet/outlet and upstream/downstream of the airflow developing zone in the duct.

#### 4.2. Analysis Method

1. Analysis model
   Simulative chamber models were placed at the same positions as in the test condition.

2. Analysis conditions
   The analysis conditions were specified as follows: simulating incompressible stationary flow without any influences of heat transfer, specifying kinematic viscosity of the air according to the temperature during the test, and using SST $k-\epsilon$ turbulent model. In addition, the same conditions were applied in the analysis with the commercial software.

3. Analysis result
As the bent pipe analysis result described in the previous section, Fig. 3 shows comparison of airflow distribution in the calculation results of OpenFOAM and the commercial software. The contours indicate that airflow distribution is equivalent in these results. For further study, air pressure losses were compared. The bar graphs in Fig. 4 show the air pressure losses calculated under two different conditions where airflow rates and duct shapes are changed. These graphs indicate that the results of OpenFOAM are equivalent to those of the commercial software. Also, comparison with the test results infers sufficient correlation can be obtained because the deviation range is only ±5%.

5. Application to HVAC Analysis

The HVAC is one of the products which have been most frequently analyzed for development of climate systems (4). Since this product mixes warm and cool air to feed comfortable wind in the cabin, CAE analyses are utilized to verify the air pressure losses and the temperature control performances. Furthermore, automotive air conditioning systems provide one ventilation mode that feeds wind from both of the VENT and FOOT positions for keeping the occupant’s head cool and feet warm. To fulfill this temperature control function, internal HVAC structures are designed with targeting an optimal discharge temperature at each outlet.

The following explains correlation studies on the HVAC performance at the VENT/FOOT mode with bench test results and calculation results of the commercial software.

5.1. Test Method

The HVAC in the vehicle has two heat exchangers (evaporator for cooling air and heater for warming) that mainly play the roles using heat sources from the engine. In this study, the test method was simplified by introducing cooled air from a low-temperature chamber to the HVAC with a blower. On the other hand, warmed air was produced through the heater core where flowing water was heated at a constant temperature. Thereby, mixture of warmed and cooled air was reproduced in the test. In addition, the discharge temperature at each HVAC outlet was measured with a thermocouple.

5.2. Analysis Method

(1) Analysis model

The analytical space was specified as an area ranging from the blower outlet to each HVAC outlet. Air mixing/distributing doors for controlling temperature characteristics were opened at the same angle as in the test condition.

(2) Analysis conditions

The conditions were specified as follows: simulating stationary flow with influences of heat convection, specifying air properties as a temperature function, using standard k-ε turbulent model, setting constant turbulent Prandtl number, applying heat transferred from the HVAC case wall as a thermal resistance in consideration of ambient temperature, and specifying functions of air pressure loss and heat generation to the heat exchangers in accordance with the test results. In addition, the same conditions were applied in the analysis with the commercial software.

(3) Analysis result

As shown in Fig. 5, the discharge temperature at each outlet has sufficient correlation between OpenFOAM
and the bench test results because the deviation range is only ±5°C. On the other hand, as Fig 6 and 7 demonstrate, the airflow velocity and the temperature distribution are equivalent in the calculation results of OpenFOAM and the commercial software.

6. Application to Windshield Defroster Analysis

The HVAC has a climate function to defrost windshields, called defroster. In product development processes, the CAE analysis is utilized to verify defroster performances based on airflow velocity in the vicinity of the windshield as an alternative indicator because the correlation between defrosting patterns and airflow velocity is generally known (5). In recent defroster designs, the CAE analysis has become more significant since the product has strict layout requirements inside the instrument panel where high-quality Head Up Display (HUD) is installed.

For this correlation studies, airflow velocity was measured in the vicinity of the internal windshield surface in vehicle simulation bench tests and compared with OpenFOAM calculation results.

6.1. Test Method

The HVAC product is located upstream of the defroster duct. However, since the test purpose was only for correlation verification, a substitutive small chamber was placed to measure the airflow velocity as shown in Fig. 8.

6.2. Analysis Model and Condition

To simulate the test conditions, a small chamber model was created, and the airflow velocity distribution around the windshield was calculated. The calculation was conducted by simulating incompressible stationary flow with SST k-ω turbulent model.

6.3. Analysis Result

Fig. 9 shows the airflow velocity at 5 mm position from the windshield. The contours in those results have the same tendency. On the other hand, Fig. 10 shows the deviation range at each measurement point between those results. The plots in the graphs indicate that
sufficient correlation can be obtained from OpenFOAM because the deviation range is only ±2 m/s.

![Fig. 9 Velocity Distribution](image)

**Fig. 9 Velocity Distribution**

![Fig. 10 Error Range of Velocity between Experiment and OpenFOAM](image)

**Fig. 10 Error Range of Velocity between Experiment and OpenFOAM**

### 7. Application to Blower Analysis in Climate System

In climate systems, there are various rotating units such as a fan, a compressor, and a centrifugal separator, all of which exchange energy with fluid. Thus, performances of these units vary by fluid forces and interacting object shapes. As an example, an operating point and efficiency of the fan significantly vary depending on the layout and the number of blades. To verify these kinds of variable characteristics, the CAE analysis is utilized.

Firstly, the application study of OpenFOAM was conducted on a centrifugal fan. This unit is used as a blower in climate systems to feed airflow with a high static pressure. In this study, inlet static pressure was calculated at a certain airflow rate with OpenFOAM and compared with the bench test results and the calculation results of the commercial software.

#### 7.1. Test Method

Airflow rate, differential pressure, and fan revolution were measured under the condition where a duct was connected to the blower outlet.

#### 7.2. Analysis Method

(1) Analysis model

As shown in Fig. 11, an open space was modeled in a certain size above the blower inlet, and an extruded duct model was connected to the outlet.

![Fig. 11 Analysis Model](image)

**Fig. 11 Analysis Model**

(2) Analysis conditions

The conditions were specified as follows: simulating incompressible stationary flow, and specifying momentum for the blade rotation with the Multiple Reference Flame (MRF) method. These conditions were also applied in the analysis with the commercial software.

(3) Analysis result

As shown in Fig. 12, the inlet static pressure in the OpenFOAM calculation result deviates approximately 10% from the bench test result but is equivalent to the calculation result of the commercial software.

![Fig. 12 Static Pressure](image)

**Fig. 12 Static Pressure**
8. Application to Motor Fan Analysis

Secondly, the application study was conducted on a motor fan (axial fan for radiator cooling) by verifying correlation of fan efficiency with the bench test results and the calculation results of the commercial software.

8.1. Test Method

A motor fan attached to a shroud was placed in a wind tunnel. Then, airflow rate (Q), static pressure difference (P), revolution (N), and torque (T) were measured. Based on the measurement results, the fan efficiency (φ) was calculated with the following equation:

\[
\phi = \frac{P \times Q}{2\pi \times N \times T}
\]

8.2. Analysis Model and Condition

To simulate the test conditions, the motor fan was attached to the shroud, and open spaces were modeled at upstream and downstream of the unit. The other conditions were the same as those in the previously-described blower analysis.

8.3. Analysis Result

The analysis was conducted at three different airflow rates. As shown in Fig. 14 and 15, the fan efficiency in OpenFOAM calculation results deviates approximately 10% from the bench test results but it tracks the fluctuation caused along the changed airflow rate. On the other hand, equivalent calculation results were obtained in OpenFOAM and the commercial software.

9. Conclusion

In this report, correlation was verified among the OpenFOAM calculation results, the bench test results, and the calculation results of the currently used commercial software on the subject of products under development with CAE analyses. The verification results conclude that OpenFOAM has equivalent accuracy to the commercial software and it is applicable in product development. A further study will be conducted to improve the analysis method for even faster processing with higher accuracy, as well as for wider applicable ranges.

Lastly, the author would like to express deep appreciation to all the contributing parties for great cooperation in this study.

References

(3) MEASUREMENT OF FLOW RATE IN SQUARE-SECTIONED DUCT BEND, JOURNAL OF THEORETICAL AND APPLIED MECHANICS 49, 2, pp.301-311, Warsaw (2011)

(4) Weimin Wang and other authors: Simulation of temperature control characteristics in the automobile A/C system, CALSONIC KANSEI TECHNICAL REVIEW Vol.5 (2008)